Fluidity Training Course

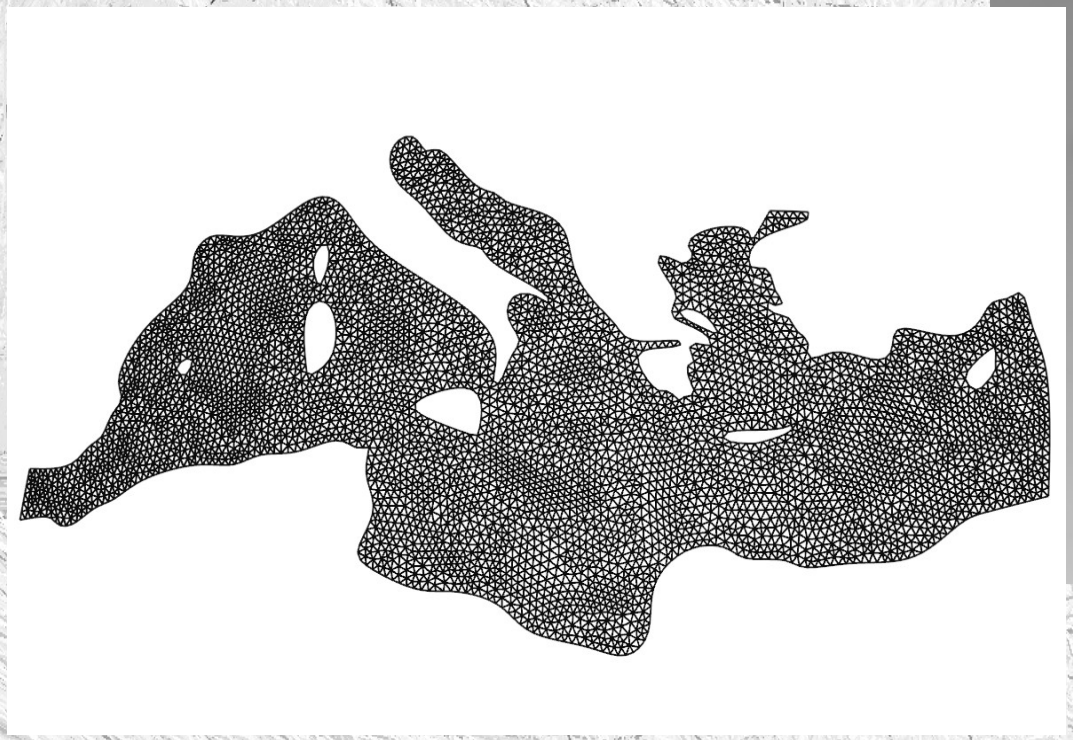
Making Meshes

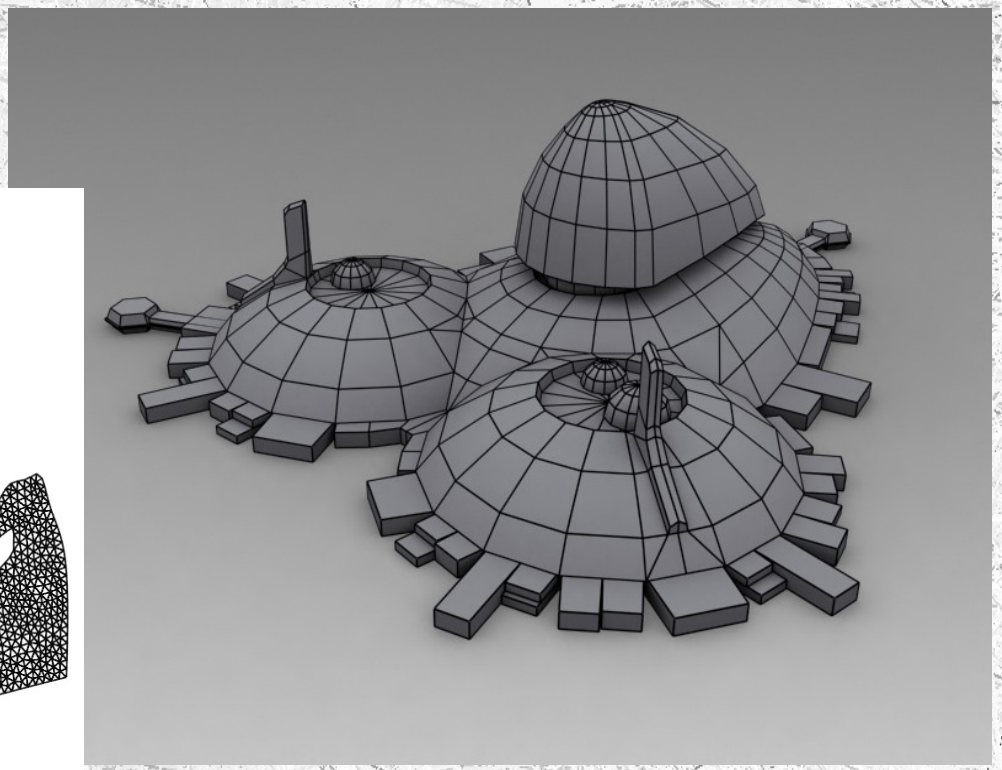
Thursday 11th November 2010

Presented by Rhodri Nelson

What is a mesh?

The discretisation of a domain into non overlapping 'elements' whose vertices are called 'nodes' and the lines connecting nodes are called 'edges', or in 3D dimensions 'faces'.





Meshing tools/packages

Gridgen (USGS): A MATLAB-based tool to construct orthongonal curvilinear grids for of NetCDF files for ECOM and SCRUM ocean circulation models.

Chimera Grid Tools (CHSSI CFD-4): A software package containing a variety of tools for the Chimera overset grid approach for solving complex configuration problems.

Channel Consulting Ltd.: TriGrid is a modelling system for building Triangular Grids. It was originally developed with oceanographic modellers specifically in mind, but is of general utility.

GMSH (Jean-Francois Remacle, Christophe Geuzaine): A Delaunay-based mesh generator, generates adapted meshes for lines, surfaces and volumes

- Freely available and easy to use!

Introduction to GMSH

GMSH tutorial

Go to http://amcg.ese.ic.ac.uk and seach for 'gmsh tutorial'
or go straight to
http://amcg.ese.ic.ac.uk/index.php?title=Local:Gmsh_tutorial

Create the following three following simple meshes:
-2D unstructured mesh (i.e. box)
-3D box (extrude the 2D box into a 3D layered box)
-2D structured mesh (for creating a periodic mesh
→ Play around with MathEval: found under 'mesh - define - fields – new'

Introduction to ocean mesh generation with GMSH
http://perso.uclouvain.be/jonathan.lambrechts/gmsh_ocean/

Inputting meshes into fluidity – Configuring the flml

Useful tools
- ***gmsh2triangle***: Convert a mesh into the triangle format
- ***triangle2vtu***: Useful for viewing your mesh

We will now go through the options in the flml and look at:-
-Setting the input triangle files
-Setting ocean boundary id's
-Extruding meshes
-Setting up a periodic mesh

The triangle format

File names end with either .node, .ele, .bound (1D), .edge (2D) or .face (3D).

node file structure is:
First line
<total number of vertices> <dimension (must be 1,2 or 3)> 0 0
Remaining lines
<vertex number> <x> [<y> [<z>]]

ele file structure is:
First line:
<total number of elements> <nodes per element>
Remaining lines:
<element number> <node> <node> <node> ... <region id>

edge file structure is:
First line:
<total number of edges> 1
Remaining lines:
<edge number> <node> <node> ... <surface id>

See the fluidity manual for further details regarding the triangle file format.